MCL 311 (Lab 2)

Ravi Shanker Meena ;   2017ME10604

1:-

Preprocessor - element type - add element - link - 3d finite stn 180

2:-

Select the key points and define the co-ordinates:

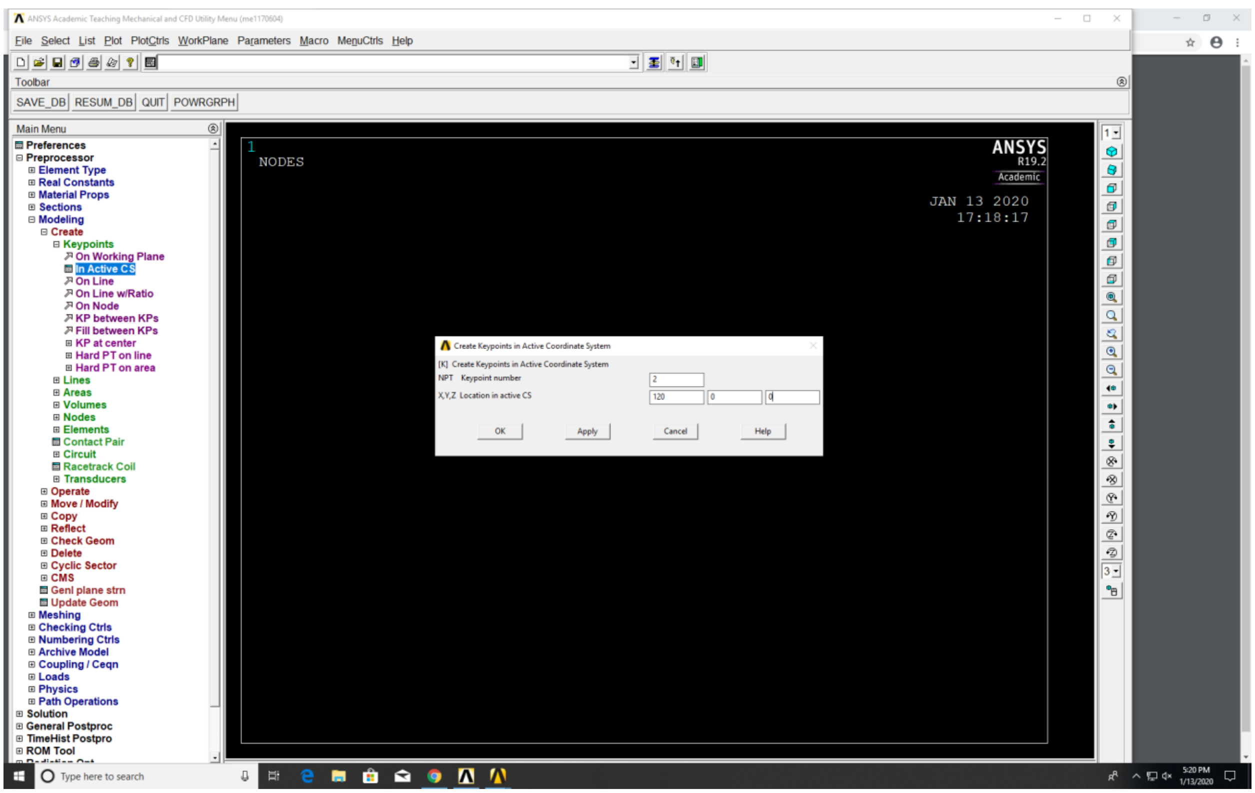
Preprocessor - modelling - create - key points - in-active CS

3:-

Preprocessor - line - in-active co-ordination - select the two key points .

1st key point coordinates - 0   0   0

2nd key point coordinates - 120   0   0



4:-

Preprocessor - modelling - sections - link - add > type “1” in section with 1D only.



5:-

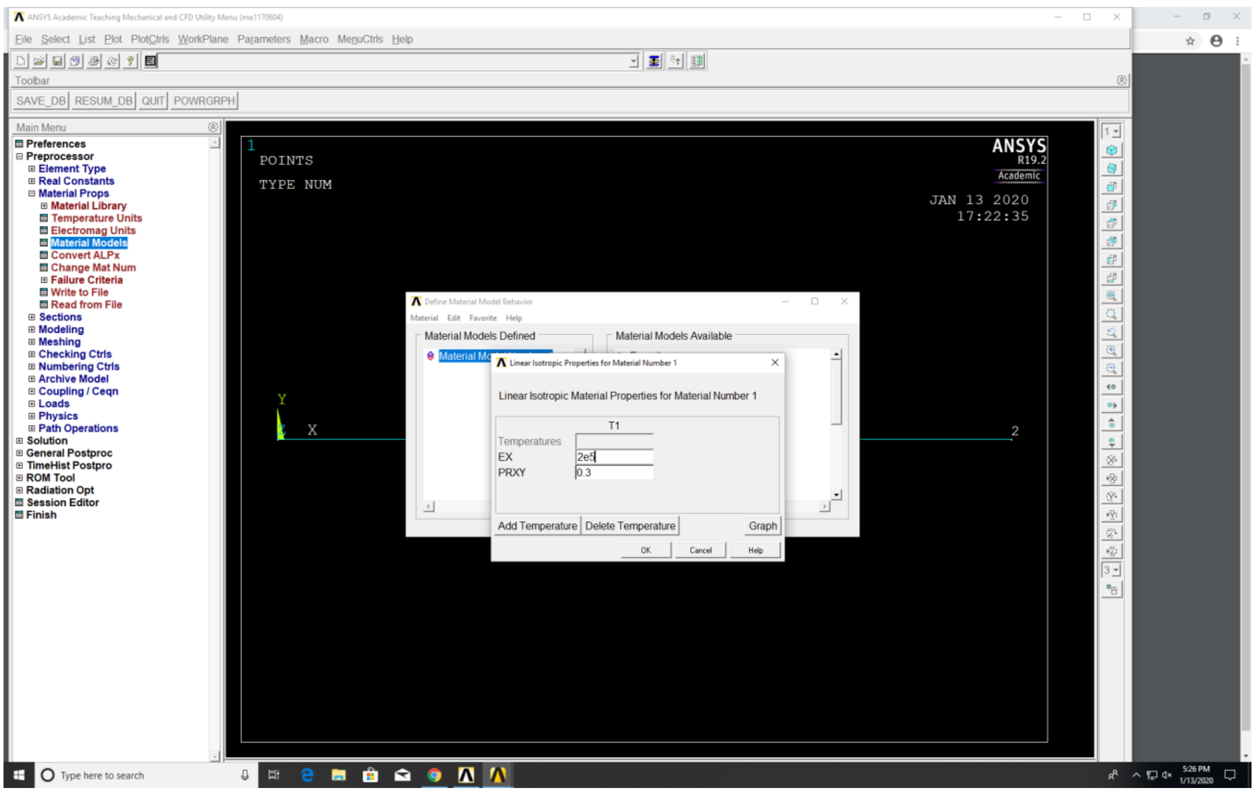
Define the material properties as per given in the problem-

Preprocessor - material properties - material models - structure - linear - elastic - isotropic

Define the material properties as given

poisson’s ratio= 0.3

young’s modulus = 2e5 MPa

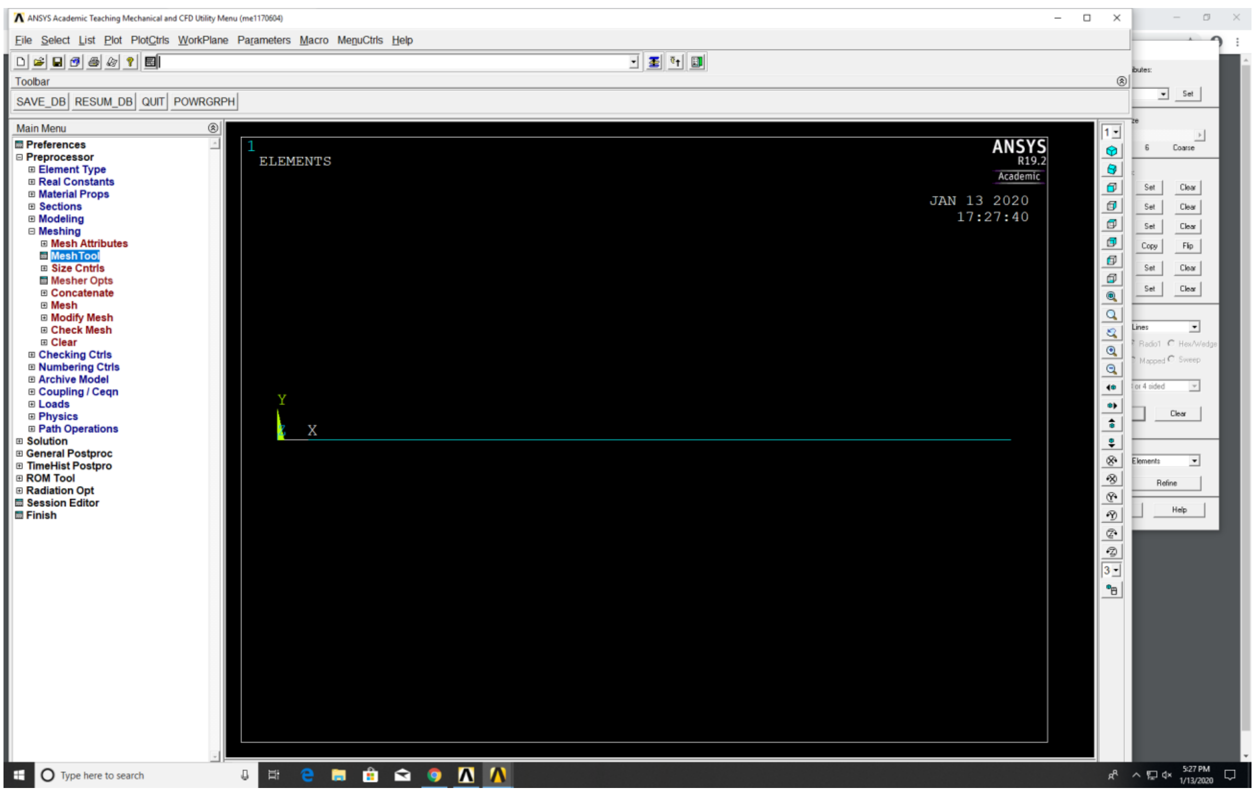


6:-

Create the meshed structure:

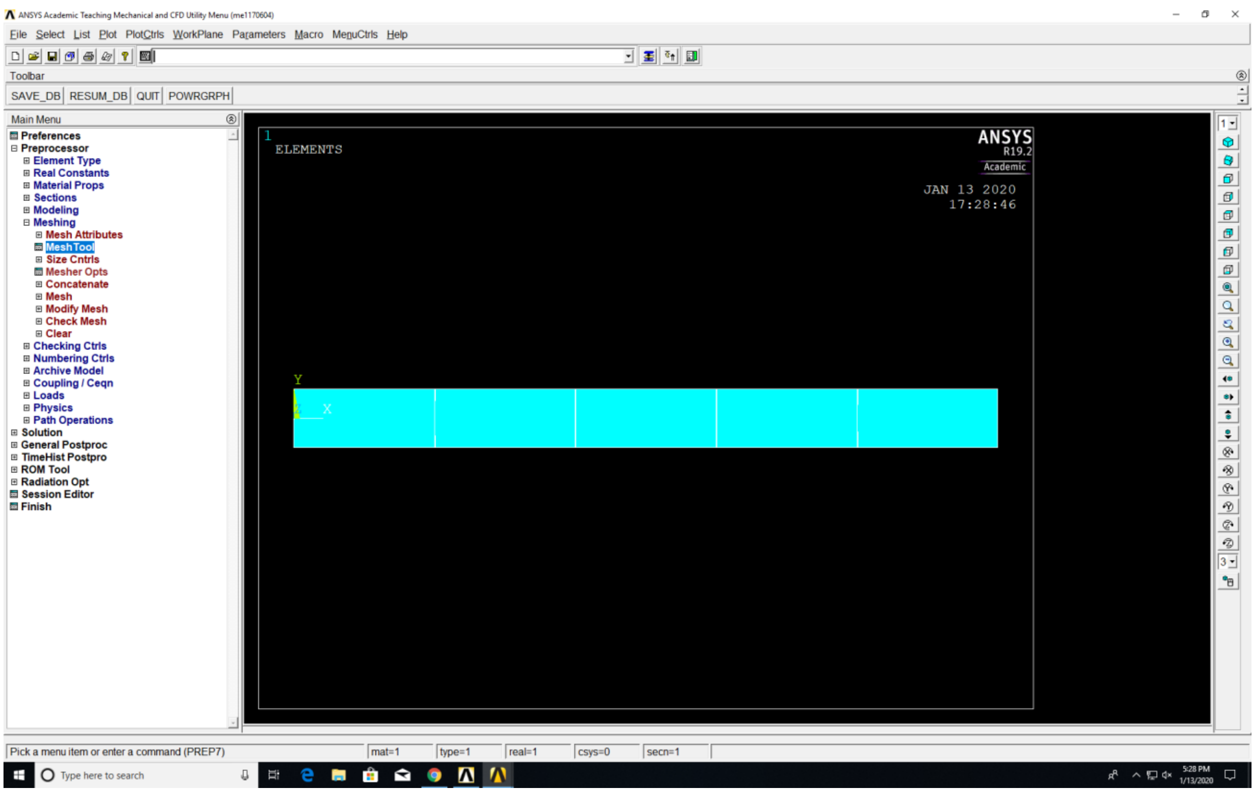
Preprocessor - meshing mesh tool - now go to global button and select the no of elements as 1,3 or 5

Finally click on the mesh button and you will be able to see your structure divided in desirable parts.



7:-

If you are not able to see your solid mechanical model then:
Go to ‘plot control’ on tool bar - style size and shape - turn ‘on’



8:-

Now apply the given load as given:

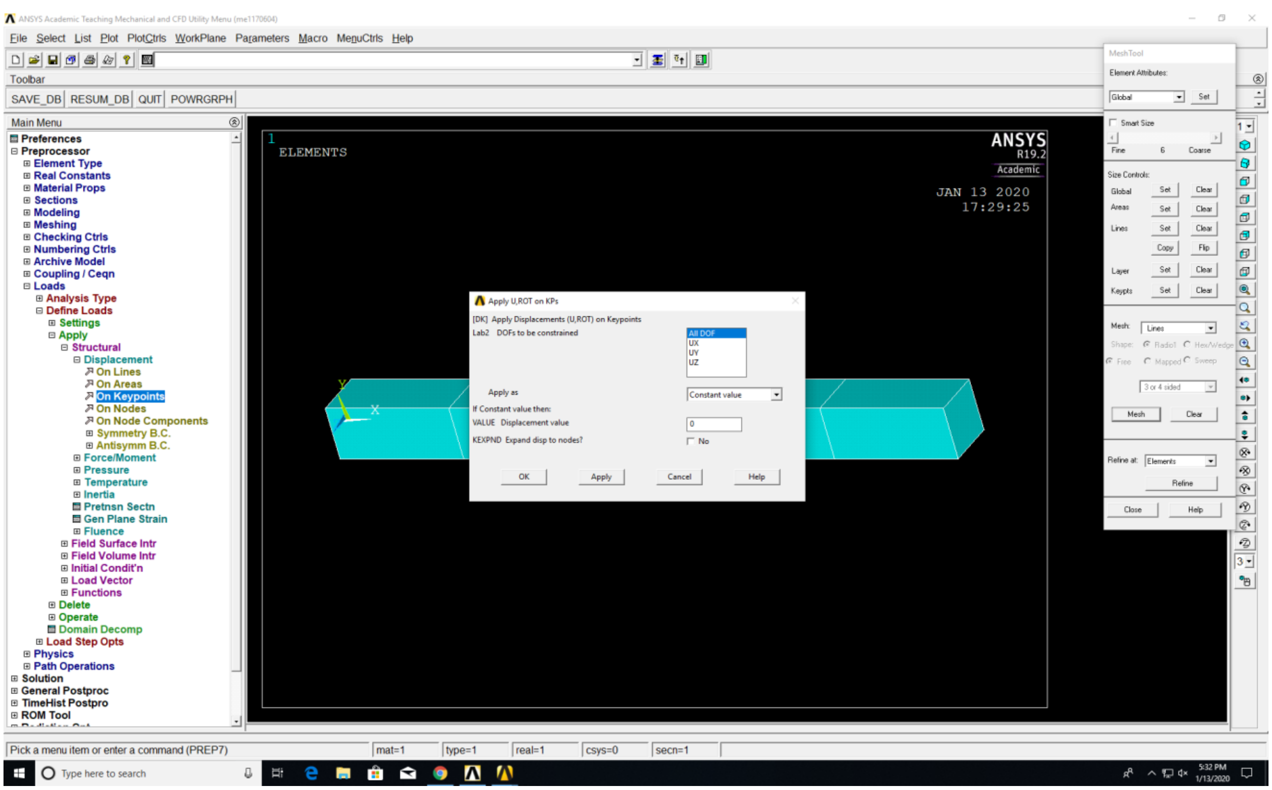
First select the key points and their degree of freedom

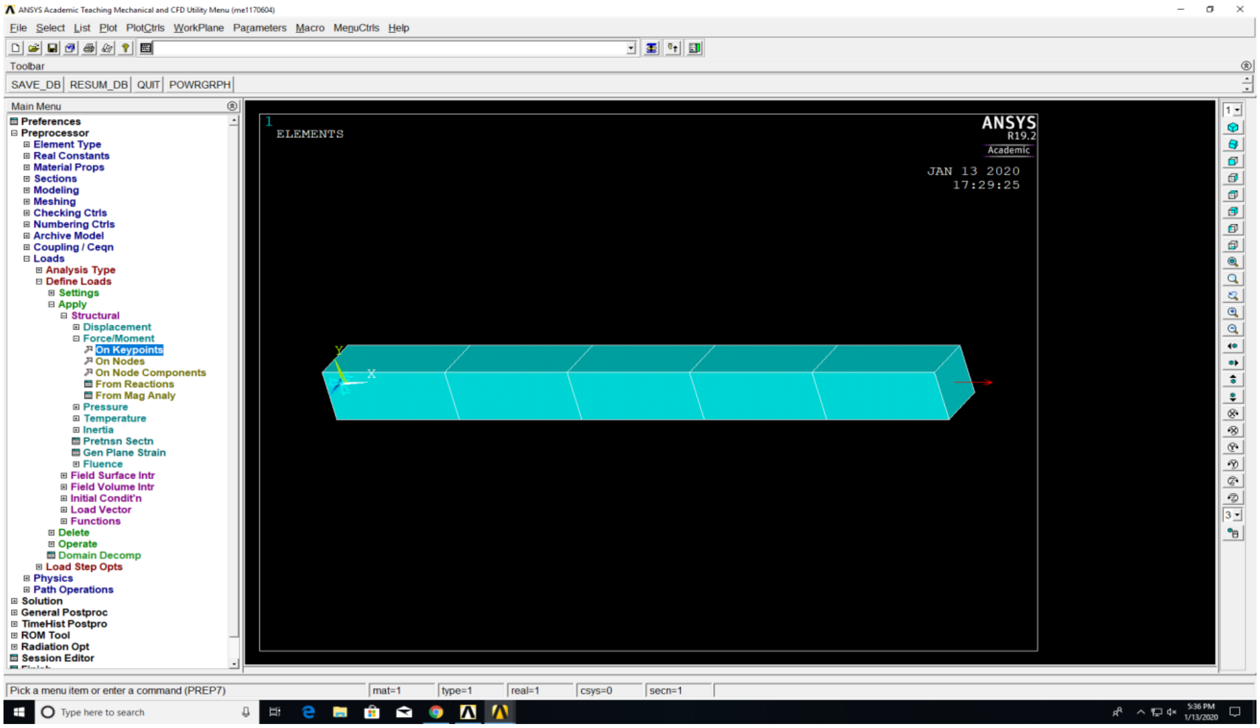
Preprocessor - loads - define load - structural - apply - structural - key point .

The enter the value of the load applied on the key point

In this case it’s 10000N

Preprocessor - loads - define load - structural - apply - force/moment - keypoint .

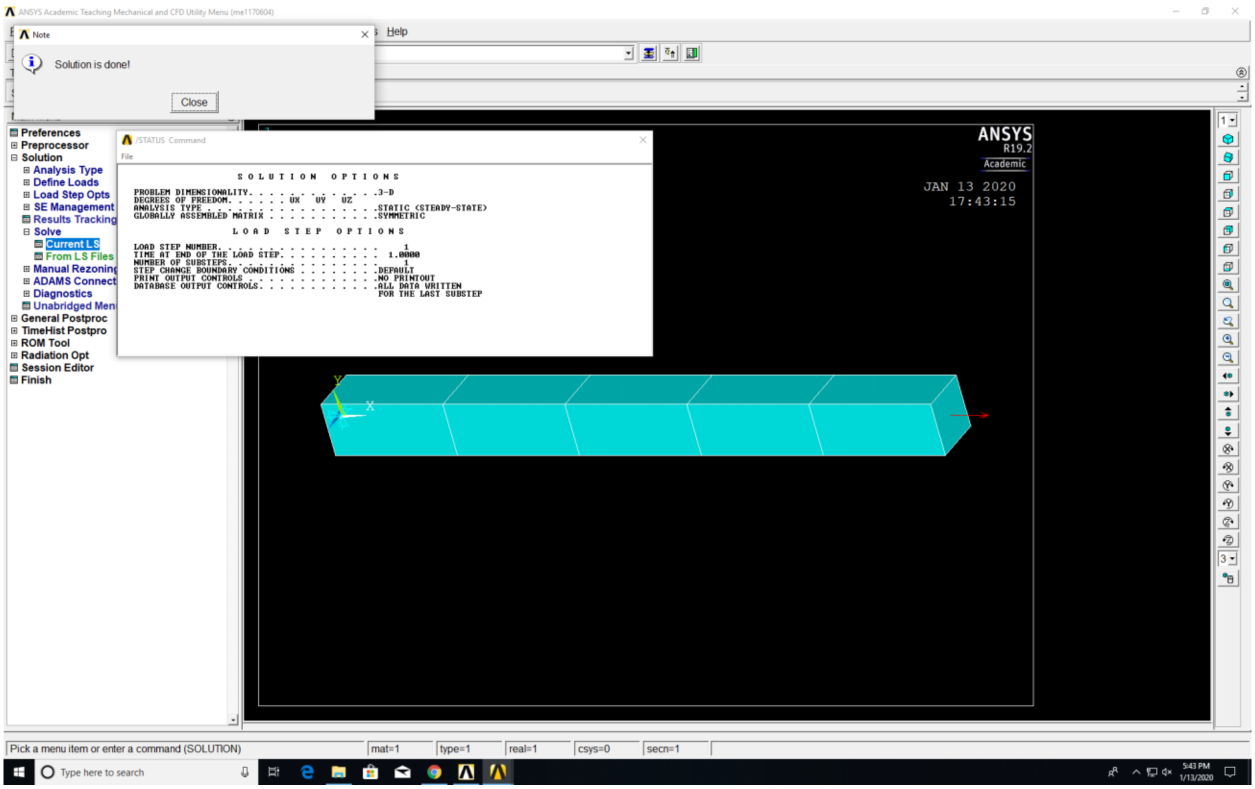



ANSYS Academic Teaching Mechanical and CFD Utility Menu (me1170604)
File Select List Plot PlotCtrls WorkPlane Parameters Macro MenuCtrls Help
Toolbar
SAVE_DB RESUM_DB QUIT POWRGRPH
Main Menu
Preferences
Preprocessor
Element Type
Real Constants
Material Props
Sections
Modeling
Meshing
Checking Ctrls
Numbering Ctrls
Archive Model
Coupling / Ceqn
Loads
Analysis Type
Define Loads
Settings
Apply
Structural
Displacement
Force/Moment
On Keypoints
On Nodes
On Node Components
From Reactions
From Mag Analy
Pressure
Temperature
Inertia
Pretnsn Sectn
Gen Plane Strain
Fluence
Field Surface Intr
Field Volume Intr
Initial Condit'n
Load Vector
Functions
Delete
Operate
Domain Decomp
Load Step Opts
Physics
Path Operations
Solution
General Postproc
TimeHist Postpro
ROM Tool
Radiation Opt
Session Editor
ELEMENTS
ANSYS
R19.2
Academic
JAN 13 2020
17:29:25
Pick a menu item or enter a command (PREP7)
mat=1
type=1
real=1
csys=0
secn=1
Type here to search
5:36 PM
1/13/2020

9:-

Go to solution - solve - current LS lines.

You will see a confirmation window showing that your solution is done .



10:-

For displacement analysis go to:

General postproc - plot results - contour plot - nodal - DOF - X-component.

For Von-mises stress analysis go to:

General postproc - plot results - contour plot - elemental - stress - von-mises stress.

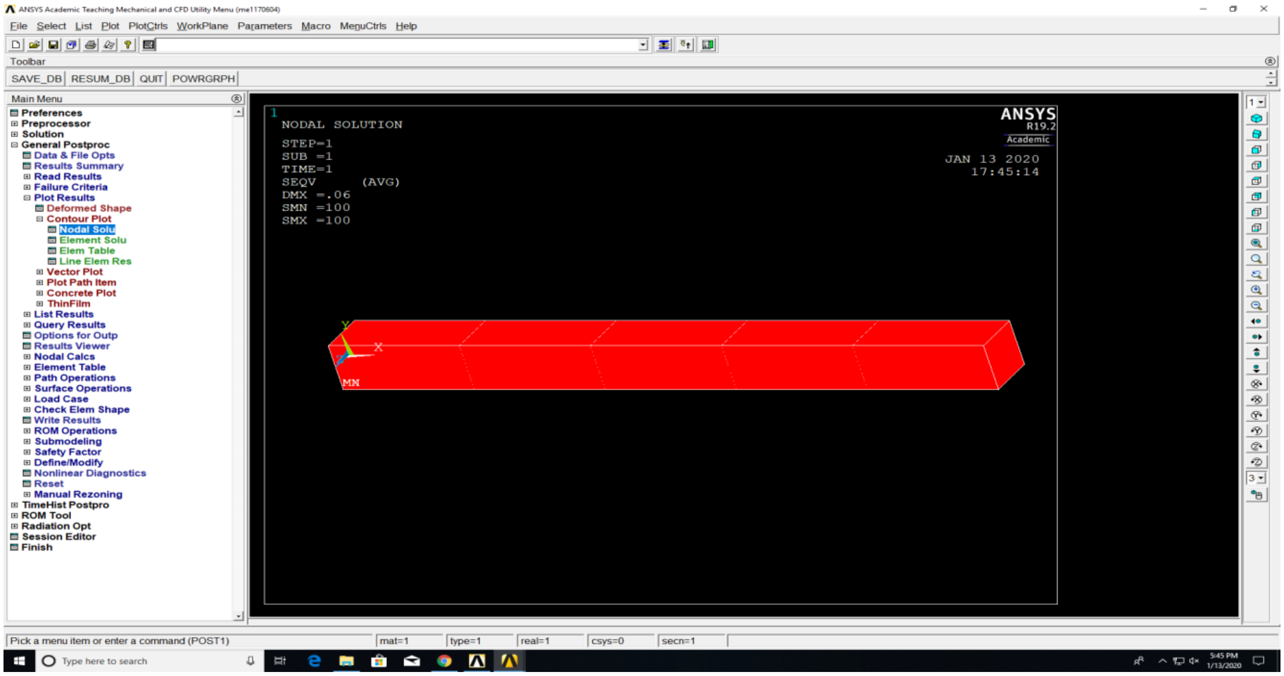
11:-

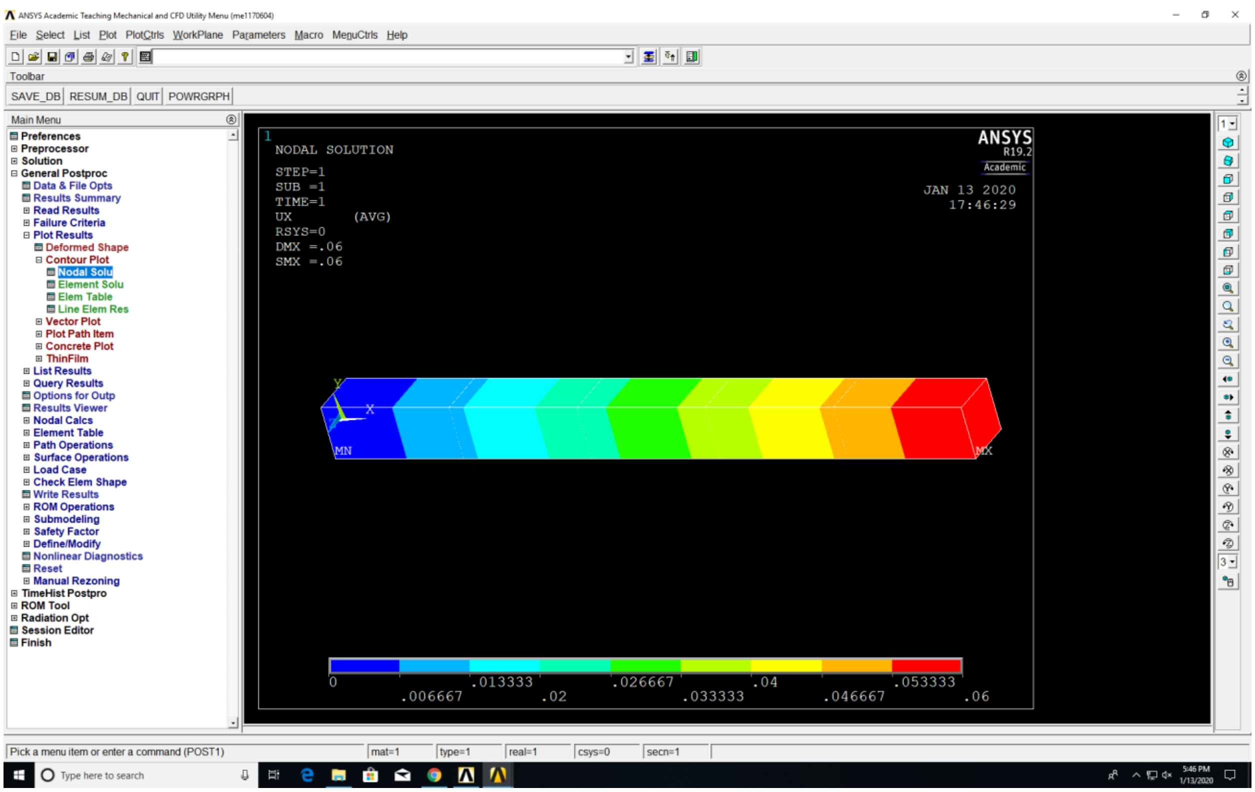
For remeshing

Go to meshing - clear - line

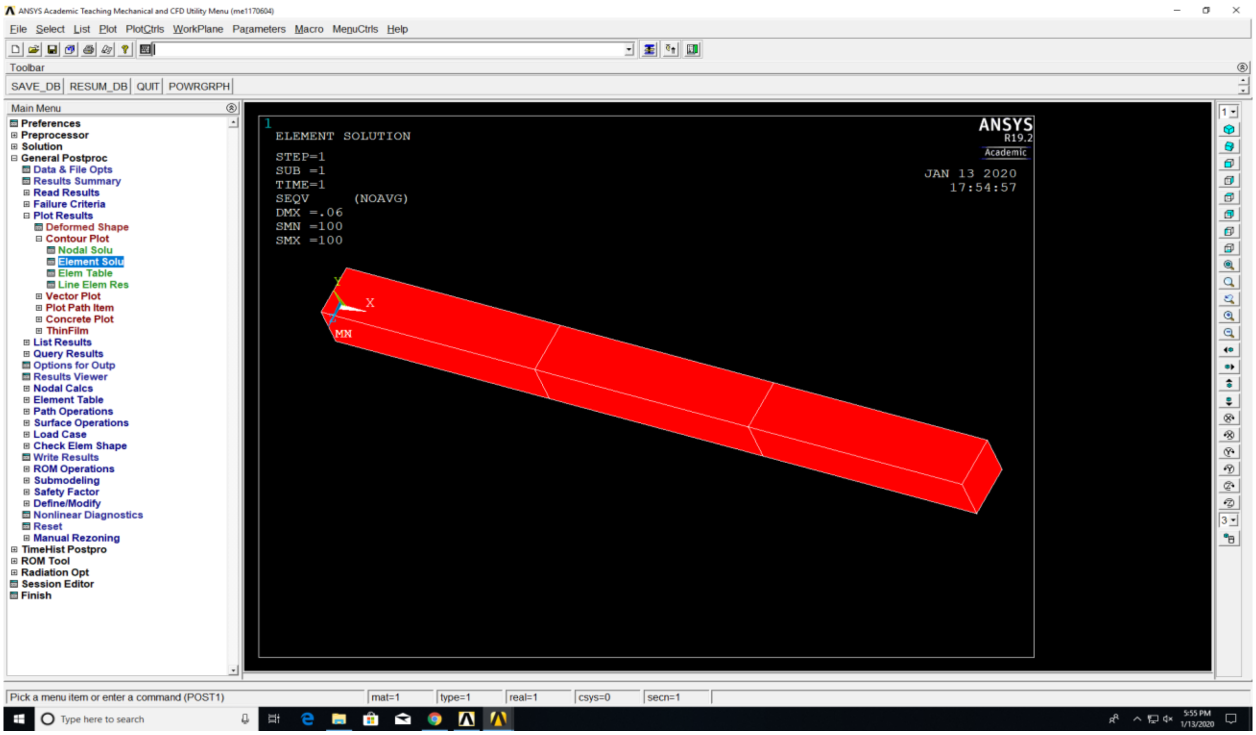
Then go to mesh tool - global - set - change the element no 5 to 3 or 1

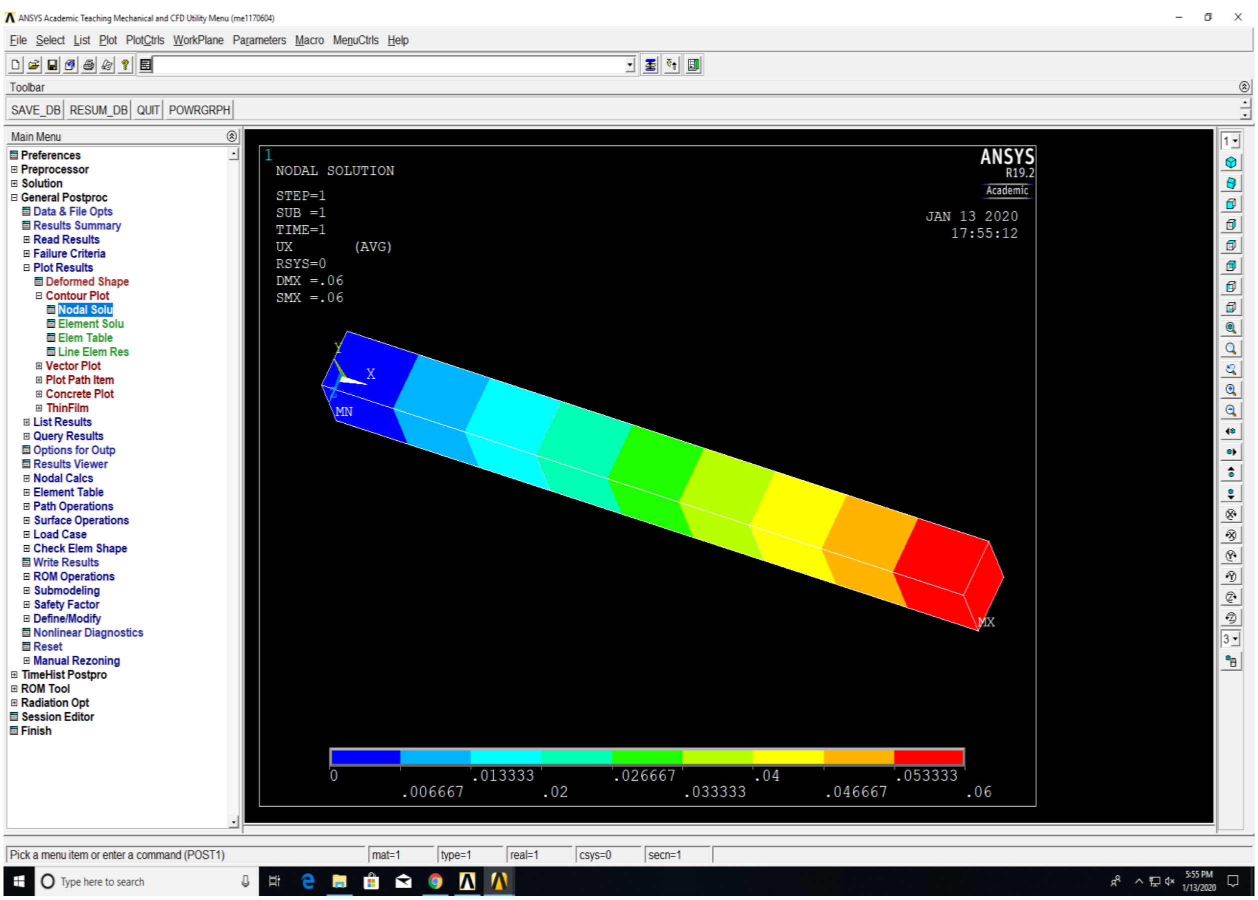
No of elements 5



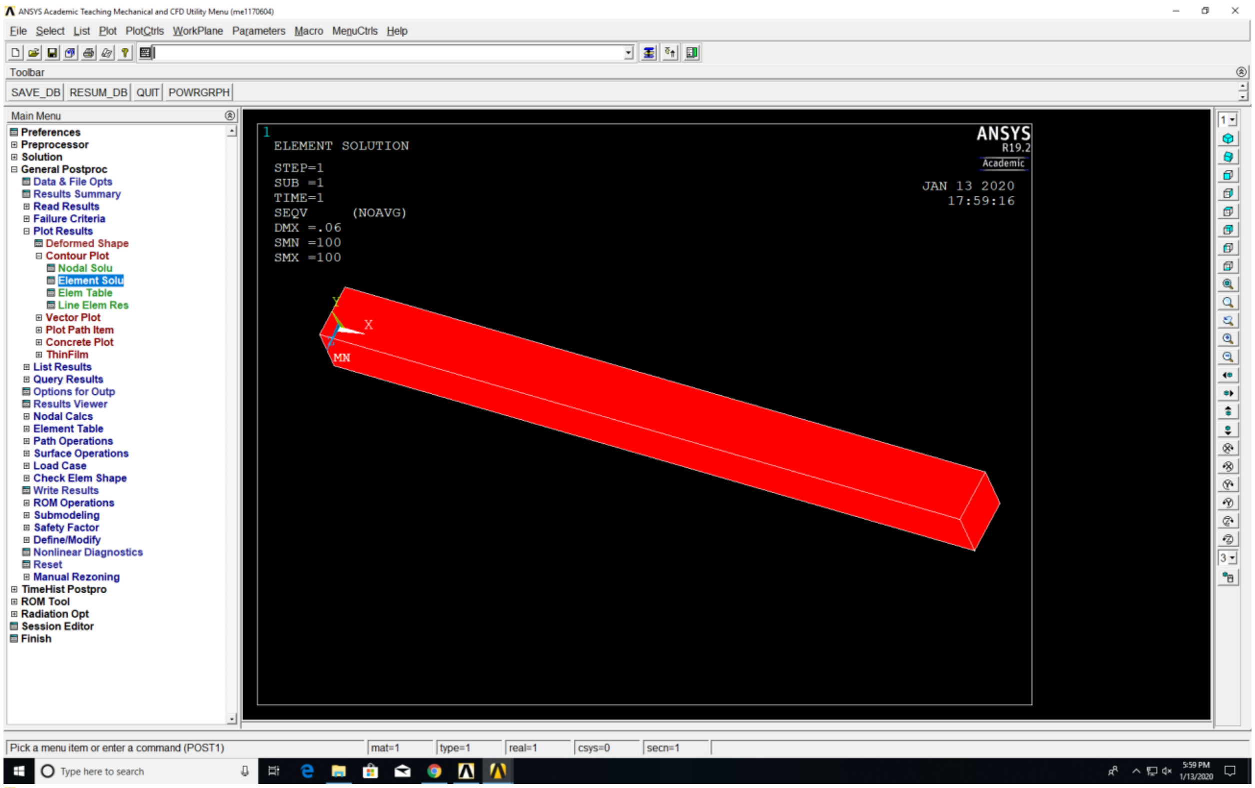


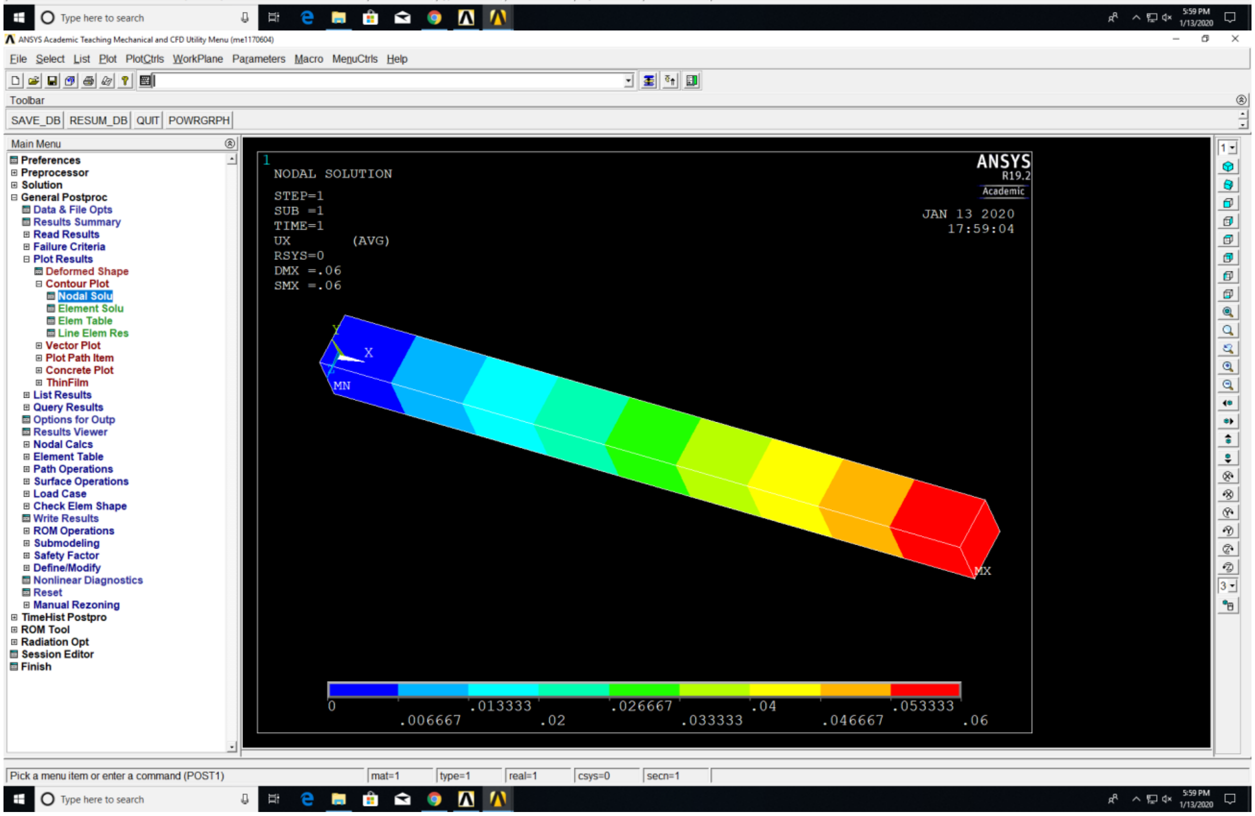
No of elements 3





No of element 1





Solution-

Solid mechanics method:

conservation of linear momentum :-

$$\frac{d\sigma_{xx}}{\partial x} = 0$$

$$\Rightarrow \quad \sigma_{xx} = C_1 = \frac{F}{A} \quad (\because \text{at } x = L)$$

$$\sigma_{xx} = \frac{F}{A}$$

$$\sigma_{xx} = \frac{10^4}{100} = 100 \text{ N/mm}^2$$

$$\epsilon_{xx} = \frac{du_x}{\partial x}$$

$$\epsilon_{xx} = \frac{\sigma_{xx}}{E} = \frac{F/A}{E} = \frac{F}{AE}$$

$$\frac{\partial u_x}{\partial x} = \frac{F}{AE} \Rightarrow \int_0^x du_x = \int_0^x \frac{F}{AE}\, dx$$

$$u_x(x) = \left(\frac{F}{EA}\right) x + C$$

B.C. 1. $u_x(x=0) = 0$

$C = 0$

therefore

$$\boxed{u_x(x) = \frac{Fx}{EA}}$$

B.C. 2. $u_x(x=L) = \frac{FL}{EA}$

max. displacement

$$\delta = \frac{FL}{EA} = \frac{10^4 \cdot 120}{200 \cdot 100} \frac{\text{N.mm}}{\text{GPa.mm}^2}$$

$$= 60 \frac{\text{N}}{\text{GPa.mm}} = 0.06 \text{ N/kPa.mm}$$

Conclusion: – both answers that we got using ANSYS & solid mechanics methods are same because the assumption we made in the solid mechanics method that it shows elasticity is true for this beam.

When we change the no of elements in meshing it does not effect our answer because the cross section on which we are applying force is same for all cases.